

**FINITE ELEMENT ANALYSIS OF 10 SPAN BEAM
BRIDGE BY USING ANSYS**

TOU SHU XIAN

**B.ENG (HONS.) CIVIL ENGINEERING
UNIVERSITY MALAYSIA PAHANG**



SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Civil Engineering (Hons.).

Signature :

Name of Supervisor : DR. CHENG HOCK TIAN

Position : Undergraduate Research Project Supervisor

Date :



STUDENT'S DECLARATION

I hereby declare that the work in this thesis is based on my original work except for quotations and citations which have been duly acknowledged. I also declare that it has not been previously or concurrently submitted for any other degree at Universiti Malaysia Pahang or any other institutions.

(Student's Signature)

Full Name : TOU SHU XIAN

ID Number : AA15287

Date : 11 January 2019

FINITE ELEMENT ANALYSIS OF 10 SPAN BEAM BRIDGE BY USING ANSYS

TOU SHU XIAN

Thesis submitted in fulfillment of the requirements
for the award of the
Bachelor Degree in Civil Engineering

Faculty of Civil Engineering and Earth Resources

UNIVERSITI MALAYSIA PAHANG

JAN 2019

ACKNOWLEDGEMENT

First and foremost, I am truly feeling appreciate and thankful to my Supervisor, Dr. Cheng Hock Tian. He had guided and helped me a lot in my research and also thesis writing. He always teaches me very patiently throughout this research and showing us the method how to use the software to run the analysis. When I face any difficulties, he will always be there ready for help. Without advice from him, I think I would be unable to complete this project report.

Besides that, I would like to thanks my family member that had gave a lot of support, understanding and encouragement to me when I am working hard to complete this research. It is a pleasure for me to thank to all my fellow friends that help me a lot for my research work. The moment that we working hard together and helping each other to solve the problem is meaningful and will always keep inside my heart.

Moreover, I am grateful to take this opportunity to thank the authority of University Malaysia Pahang (UMP) for providing me a chance and platform for gaining knowledge. Besides, thanks to all lectures of Civil Engineering & Earth Resources faculty. Thank you very much.

Finally, again I would like to express my thanks to all who involve in completing this research. Thank you so much.

ABSTRAK

Jambatan balok adalah salah satu daripada jenis jambatan yang paling biasa digunakan kerana strukturnya mudah untuk dibina dan diselenggara. Oleh itu, mereka lebih disukai berbanding jambatan jenis lain apabila ia memberikan penyambungan dalam jarak dekat. Dalam pembinaan, jurutera boleh menghubungkan sejumlah jambatan jambatan bersama untuk mewujudkan jambatan yang lebih lama. Dalam kajian ini, jambatan balok dianalisis dengan menggunakan pengiraan manual dan juga reka bentuk dalam perisian ANSYS. Terdapat banyak kaedah yang boleh digunakan untuk menganalisis kelakuan jambatan rasuk. Walau bagaimanapun, di antara semua kaedah ini, kaedah unsur terhingga dan kaedah probabilistik akan menjadi kaedah yang sangat berkesan untuk mendapatkan kekuatan dan tingkah laku jambatan rasuk. Tujuan utama analisis ini adalah menentukan tingkah laku jambatan balok dengan menggunakan perisian ANSYS. Gred konkrit bahagian HA35 dan keluli bertetulang yang digunakan di bahagian EHE S500. Nilai-nilai bagi pemboleh ubah masukan dijana secara rawak dengan menggunakan Simulasi Monte Carlo atau sebagai contoh yang ditetapkan menggunakan Kaedah Surface Response. Sebanyak 1000 simulasi telah dibuat untuk memastikan analisisnya lebih tepat dan tepat. Melalui rangsangan, kajian ini dapat membuktikan bahawa keadaan jambatan balok di bawah kewujudan rawak dan ketidakpastian boleh dianalisis dengan menggunakan analisis elemen terhingga probabilistik menggunakan ANSYS.

ABSTRACT

Beam bridges are one of the most familiar used types of bridge as structurally they are simple to construct and maintain. Hence, they are preferred over other types of bridges when it comes to providing connectivity within short distance. In construction, engineer can link a number of spans of beam bridge together to create a longer bridge. In this study, the beam bridge was analyzed by using manual calculation and also design in ANSYS software. There are numerous methods can be used to analyses the behavior of the beam bridges. Nevertheless, among all these methods, finite element method and probabilistic method will be a very effective method used to obtain the strength and behavior of the beam bridges. The main purpose of this analysis is to determine the behavior of the beam bridge by using ANSYS software. The concrete grade of the section is HA35 and the reinforced steel used in the section is EHE S500. The values for the input variables are generated randomly by using Monte Carlo Simulation or as prescribed samples using Response Surface Methods. A total number of 1000 of simulations had been made to make sure the analysis is more precise and accurate. Through the stimulation, this study can prove that the condition of the beam bridge under the existence of randomness and uncertainty can be analyzed by apply probabilistic finite element analysis using ANSYS.

TABLE OF CONTENTS

SUPERVISOR’S DECLARATION	ii
STUDENT’S DECLARATION	iii
DECICATION	v
ACKNOWLEDGEMENTS	iv
ABSTRACT	vi
ABSTRAK	vii
TABLE OF CONTENTS	viii
LIST OF TABLES	xii
LIST OF FIGURES	xiii
LIST OF SYMBOLS	xvii
LIST OF ABBREVIATIONS	xviii

CHAPTER 1 INTRODUCTION

1.1	Introduction	1
1.2	Problem Statement	2
1.3	Objective	2
1.4	Scope of Study	3
1.5	Expected Outcome	3
1.6	Significance of Study	3

CHAPTER 2 LITERATURE REVIEW

2.1	Introduction to Beam Bridge	4
2.2	ANSYS Program	5
2.2.1	Introduction	5
2.3	Probabilistic Analysis in Finite Element Modeling	5
2.4	Probabilistic Design System	6
2.5	Monte Carlo Simulation and Latin Sampling	7

CHAPTER 3 METHODOLOGY

3.1	Introduction	8
3.2	Preprocessing: Defining the Problem	9
3.2.1	Modelling 10 span Beam Bridge	9
3.2.2	Entering Title	10
3.2.3	Defining CivilFEM	10
3.2.4	Defining Beam Element Types for Code Checking	12
3.2.5	Defining Material Properties	13
3.2.6	Defining Cross sections and Member Properties	13
3.2.7	Defining Reinforcements properties for Top and Bottom bar	14
3.2.8	Defining Member Properties	16
3.2.9	Entering Nodes	16
3.2.10	Setup all Force and Support	18
3.2.11	Result by Load Step Method	21
3.2.12	Reaction Forces	22
3.2.13	Deflection	23
3.2.14	Read Result	25
3.2.15	Force & Moment	25
3.2.16	Result by Eurocode	28

3.2.17 Force and Moment List by Load Step	30
---	----

CHAPTER 4 RESULTS AND DATA ANALYSIS

4.1	Introduction	34
4.2	Probabilistic Results from ANSYS	35
4.2.1	Random Input Variables	35
4.2.2	Statistics of the Probabilistic Results	41
4.2.3	Sample History Plots	42
4.2.4	Histogram Plots of Input Parameters	44
4.2.5	Histogram Plots of Output Parameters	49
4.2.6	Cumulative Distribution Function Plots	50
4.2.7	Sensitivity Plots	55
4.2.8	Linear Correlation Coefficients	57
4.2.9	Spearman Rank Order Correlation Coefficients	58

CHAPTER 5 CONCLUSIONS AND RECOMMENDATIONS

5.1	Introduction	61
5.2	Conclusion	61
5.3	Recommendation	62

REFERENCES	63
-------------------	----

APPENDICES

A1	Eurocode Checking of Bridge	64
B1	Log File	66

LIST OF TABLES

Table No.	Title	Page
3.0	Beam Element Force and Moment	31
3.1	Stress and Strain	33
4.1	Random Input Variables Specifications	35
4.2	Statistical of Random Input Variables	42
4.3	Statistical of Random Output Variables	42
4.4	Linear Correlation Coefficients between Input Variables	57
4.5	Linear Correlation Coefficients between Input and Output Variables	58
4.6	Linear Correlation Coefficients between Output Variables	58
4.7	Spearman Rank Order Correlation Coefficients between Input Variables	59
4.8	Spearman Rank Order Correlation Coefficients between Input And Output Variables	59
4.9	Spearman Rank Order Correlation Coefficients between Output Variables	60

LIST OF FIGURES

Figure No.	Title	Page
2.1	Typical Type of Beam Bridge	4
3.0	Flow Chart of The Research Process	9
3.1	The title for the model	10
3.2	Activate CIVILFEM	10
3.3	CIVILFEM Setup Option	11
3.4	Setting up Units	12
3.5	Beam element types for code checking	12
3.6	Material that chooses for the structure	13
3.7	Cross section for beam	14
3.8	Reinforcement Properties for the top bar	15
3.9	Reinforcement Properties for the bottom bar	15
3.10	Beam and shell properties for the section	16
3.11	Creates of nodes in active CS	17
3.12	Nodes that done establish	17
3.13	Elements that has been finish joint	18
3.14	The displacement on nodes	18
3.15	the ploard files will be choosing in the file	19
3.16	pload.mac	19
3.17	the loading that moves along the span	20
3.18	the forces and moments applied at 'Mz' direction on the structure	20
3.19	the results for the current LS solve	21
3.20	Load step number	21

3.21	Graph Force and Moment Results	22
3.22	Plot Deformed Shape	22
3.23	Deformed Shape	23
3.24	Contour Nodal Solution Data	24
3.25	Contour Plot of Deflection	24
3.26	Read Results by Load Step Number	25
3.27	Graph Force and Moment Results	26
3.28	Axial Force Diagram	26
3.29	Shear Force Diagram	27
3.30	Bending Moment Diagram	27
3.31	the output for 3D axial + bending checking	28
3.32	the checking for the cracking	29
3.33	the graph for check cracking	29
3.34	graph for axial + bending checking	30
3.35	List Force and Moment	30
3.36	List Stress & Strain Result	33
4.1	PDF & CDF of Input Random Variable EX1	36
4.2	PDF & CDF of Input Random Variable maximum strain	37
4.3	PDF & CDF of Input Random maximum stress	38
4.4	PDF & CDF of Input Random Variable MZ1	39
4.5	PDF & CDF of Input Random Variable MZ2	40
4.6	PDF & CDF of Input Random Variable Poison	41
4.7	Mean Value History for Output Parameter POISSON2	43
4.8	Simulation Sample Values for Output Parameter EX1	44

4.9	Histogram of Input Parameter for EX1	44
4.10	Histogram of Input Parameter for MAXIMUM STRAIN	45
4.11	Histogram of Input Parameter for Maximum stress	46
4.12	Histogram of Input Parameter for MZ1	47
4.13	Histogram of Input Parameter for MZ2	48
4.14	Histogram of Input Parameter for POISSON1	49
4.15	Histogram of Input Parameter for Maximum Deflection	50
4.16	CDF of Input Variable EX1	51
4.17	CDF of Input Variable MAXIMUM STRAIN	52
4.18	CDF of Input Variable MAXIMUM STRESS	52
4.19	CDF of Input Variable MZ1	53
4.20	CDF of Input Variable MZ2	54
4.21	CDF of Input Variable POISSON 1	54
4.22	CDF of Output Variable MAXIMUMDEFLECTION	55
4.23	Sensitivity Plot for MZ3	56

LIST OF SYMBOLS

LIST OF ABBREVIATIONS

2D	Two Dimensional
CIVIFEM	Civil Finite Element Method
LatBuck	Lateral Buckling
ChckAxis	Check Axis
BMSHPRO	Beam and Shell Properties
CS	Coordinate System
LS	Load Step
DOF	Degree of Freedom
PRES	Pressure
GAUS	Gaussian
POISON	Poison ratio
PDF	Probabilistic density function
CDF	Cumulative distribution function
MAXIMUMDEFLECTION /MAX_DEFLECTION	Maximum Deflection

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Nowadays, the main purpose to conduct construction of the bridge in a city is to smoothen the traffic flow in a city that has high population. Other than that, bridges are used to connect one city to another. Bridges form important part of a nation's infrastructure. Maintenance and upgrading of these bridges are costly in terms of both the direct repair costs and the economic loss arising from traffic delays during repairs. In the assessment of any highway bridge structure, traffic loading is one of the most variable factors. While approaching the problem of live loads for bridges, it is necessary to make a comparison between the maximum moments in different bridge members as obtained from the codes. (Shoukry,M.E. , 2017)

Although traffic contents are important information, bridge load carrying capacity is more influenced by the effect that loading cause on the structure. Traffic load is a dynamic load hence it is important to understand dynamic behavior and possible effects from moving vehicles. The dynamic load depends on various criteria like: vehicle type, vehicle weight, axle configuration, bridge material, bridge span length, road roughness and transverse position of the truck on the bridge. (Paeglitea,I. , 2017)

An influence lines for a specific response such as axial force, reaction, shear force, and bending moment it meant as a diagram in which the ordinate at any point equals the value of that response attributable to a unit load acting at that point on the structure. In the engineering study the influence line presents the variety of capacity. Every element

for the structure of a bridge is designed according to the most severe conditions that can be determined by using suitable method known as 'influence line'. Influence lines of responses statically determinate structures contain only.

1.2 PROBLEM STATEMENT

On the evening of 31st July 1988, the unthinkable happened when the Sultan Abdul Halim Ferry Terminal bridge in Butterworth, Penang suddenly collapsed. Thousands of people celebrating the festivities moved from Butterworth to Penang Island on this day, causing huge crowds to gather at the jetty, and numbers went up during the evening. It was reported that there were about 10,000 people standing on the bridge before tragedy struck. At around 4.40pm, the floor beams beneath suddenly snapped under the pressure of all those people, causing the floorboards to collapse and the passengers to fall. Many of them were crushed underneath the concrete columns and floorboards. (Tara Thiagarajan, 2018).

From this incident, it clearly shows that the quality of the construction must be always fit with the loading that is acting on the structure. To maintain the quality and safety, the study about the bridge must be propose to get the exact or accurate result in design.

1.3 OBJECTIVES

Research objectives are an important part for every project or thesis in order to conduct a good research. It acts as guidance for researcher to achieve the final objective and avoid researcher to digress. The objective for the research play an important role for researcher to understand and remind themselves of the criteria and testing that should be done in a research. The main objectives of this study are:

- I. To plot and analyze the influence line by using reciprocity theorem at the left mid span of the bridge beam.
- II. To determine the force and moment, stress & strain and deflection on the beam using ANSYS software.

III. To make sure the beam passes all code checking.

1.4 SCOPE OF STUDY

In this research, finite element method (FEM) model is developed to stimulate the behavior of the structure from linear through nonlinear response and up to failure, using the ANSYS program. Modeling simplifications and assumptions developed during this research are presented.

The probabilistic analysis approach is applied to account for the variability in fabrication. Probabilistic methodology applied in finite element analysis provides another alternative way of structural analysis of the beam of the bridge to achieve a robust and reliable design in a most efficient way. The task that comprising the research program is conduct Monte Carlo simulation and Response Surface simulation to analyze the effect of parameter uncertainty for the beam of the bridge.

1.5 EXPECTED OUTCOME

This research claims to find out the behaviors and passing code checking of every span of the bridge. The behaviors are deformation, deflection axial stress and strain checking according to Eurocode 2.

1.6 SIGNIFICANCE OF STUDY

This research will provide the information of maximum deformation, deflection axial stress and strain through the software. It is really good to practice to analyze the structure and make sure it is safe before constructing it. Furthermore, the element of the bridges is always design by using the most severe condition. By using the software, we can do the stimulation and found out the maximum deflection and thus we can obtain more detail information regarding the structure.

REFERENCES

- BS EN 1991-1-1: (2002). *Eurocode 1: Actions on structures. General actions. Densities, self-weight, imposed loads for buildings*, BSI.
- G Lackshmi, N. (2009). *Finite Element Analysis*. BS Publications.
- Mohamed, H.-Z., David, W., & M., A. (1983). Nonlinear finite element analysis. *J. Struct. Eng.*, 353-368.
- University of Alberta - ANSYS Tutorials*. (2001). Retrieved from <http://www.mece.ualberta.ca/tutorials/ansys/>
- G Lackshmi, N. (2009). *Finite Element Analysis*. BS Publications.
- Gao, L., & Zeng, X. Y., & Wang, Y. (2011). Research on Several Problems about Nonlinear Analysis of RC Beams Based on ANSYS. *Journal of Water Resources and Architectural Engineering*. 92-96
- Scheer, J. (2010). *Failed Bridges Case Studies, Causes and Consequences*. Berlin: Ernst & Sohn
- Karthiga @Shenbagam.N. Preetha.V. and Jayaraman.A. 2014. Finite element analysis of reinforced concrete beams. *International Journal of Innovative Research in Advanced Engineering (IJIRAE)*. **1(8)**: 272-285.
- T. Subramani. (2014). Crack identification in reinforced concrete beams using Ansys software. *Int. Journal of Engineering Research and Applications*. **4(6)**:133-141
- Shoghijavan.M. and Starossek.U. (2018). An analytical study on the bending moment acting on the girder of a long span cable-supported bridge suffering from cable failure. *Engineering Structures*. **167**: 166-174.
- Wang,N.B. He,L.X. Ren,W.X. Huang,T.L. (2017). Extraction of influence line through a fitting method from bridge dynamic response induced by a passing vehicle. *Engineering Structures*.**151**:648-664.
- University of Alberta - ANSYS Tutorials*. (2001). Retrieved from <http://www.mece.ualberta.ca/tutorials/ansys/>